2026 R1

EMPOWER INNOVATORS TO DRIVE HUMAN ADVANCEMENT

2026 R1

Getting Started with IcepakFEA: Modal Solution – Coil

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
https://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2026 R1
January 2026

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001:2015 companies.

Conventions Used in this Guide

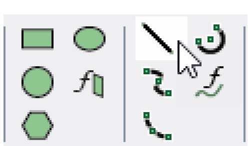
Please take a moment to review how instructions and other useful information are presented in this documentation.

- Procedures are presented as numbered lists. A single bullet indicates that the procedure has only one step.
- Bold type is used for the following:
    - Keyboard entries that should be typed in their entirety exactly as shown. For example, “**copy file1**” means you must type the word **copy**, then type a space, and then type **file1**.
    - On-screen prompts and messages, names of options and text boxes, and menu commands. Menu commands are often separated by greater than signs (>). For example, “click **HFSS** > **Excitations** > **Assign** > **Wave Port**.”
    - Labeled keys on the computer keyboard. For example, “Press **Enter**” means to press the key labeled **Enter**.
- Italic type is used for the following:
    - Emphasis.
    - The titles of publications.
    - Keyboard entries when a name or a variable must be typed in place of the words in italics. For example, “**copy** *filename*” means you must type the word **copy**, then type a space, and then type the name of the file.
- The plus sign (+) is used between keyboard keys to indicate that you should press the keys at the same time. For example, “Press Shift+F1” means to press the **Shift** key and, while holding it down, press the **F1** key also. You should always depress the modifier key or keys first (for example, Shift, Ctrl, Alt, or Ctrl+Shift), continue to hold it/them down, and then press the last key in the instruction.

Accessing Commands: *Ribbons, menu bars,* and *shortcut menus* are three methods that can be used to see what commands are available in the application.

- The *Ribbon* occupies the rectangular area at the top of the application window and contains multiple tabs. Each tab has relevant commands that are organized, grouped, and labeled. An example of a typical user interaction is as follows:

  "Click **Draw** > **Line**"

This instruction means that you should click the **Line** command on the **Draw** ribbon tab. An image of the command icon, or a partial view of the ribbon, is often included with the instruction.

- The *menu bar* (located above the ribbon) is a group of the main commands of an application arranged by category such File, Edit, View, Project, etc. An example of a typical user interaction is as follows:

  "On the **File** menu, click the **Open Examples** command" means you can click the **File** menu and then click **Open Examples** to launch the dialog box.

- Another alternative is to use the *shortcut menu* that appears when you click the right-mouse button. An example of a typical user interaction is as follows:

  “Right-click and select **Assign Excitation** > **Wave Port**” means when you click the right-mouse button with an object face selected, you can execute the excitation commands from the shortcut menu (and the corresponding sub-menus).

Getting Help: Ansys Technical Support

For information about Ansys Technical Support, go to the Ansys corporate Support website, http://www.ansys.com/Support. You can also contact your Ansys account manager in order to obtain this information.

All Ansys software files are ASCII text and can be sent conveniently by e-mail. When reporting difficulties, it is extremely helpful to include very specific information about what steps were taken or what stages the simulation reached, including software files as applicable. This allows more rapid and effective debugging.

Help Menu

To access help from the Help menu, click **Help** and select from the menu:

- **[product name] Help** - opens the contents of the help. This help includes the help for the product and its *Getting Started Guides*.
- **[product name] Scripting Help** - opens the contents of the *Scripting Guide*.
- **[product name] Getting Started Guides** - opens a topic that contains links to Getting Started Guides in the help system.

Context-Sensitive Help

To access help from the user interface, press **F1**. The help specific to the active product (design type) opens.

You can press **F1** while the cursor is pointing at a menu command or while a particular dialog box or dialog box tab is open. In this case, the help page associated with the command or open dialog box is displayed automatically.

Table of Contents

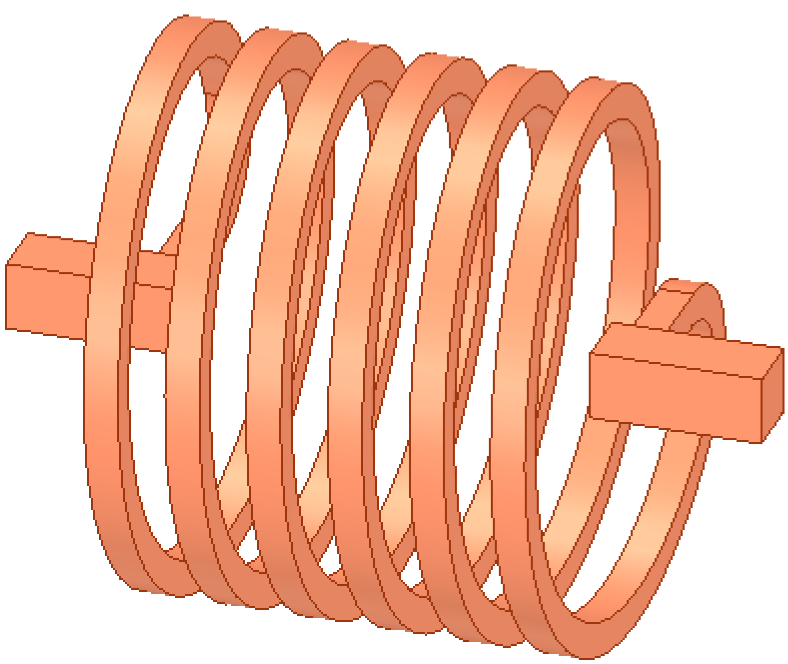
1 - Introduction

In this *Getting Started* guide, you will learn how to determine the natural frequencies of vibration, mode shapes, and mass participation using the *Ansys Electronics Desktop* application. Specifically, the guide provides an example of setting up, solving, and reviewing the results of a *Modal* solution in an *IcepakFEA* design. The focus is on modal analysis and not solid modeling. Therefore, a CAD solid model has been provided as the starting point for the exercise.

By following the procedures in this guide, you will learn how to perform the following tasks:

- Insert an IcepakFEA design and choose the Modal solution type
- Set 3D UI options and choose the model units
- Import a CAD solid model
- Select the material
- Apply fixed support boundaries
- Set up, validate, and solve the modal analysis
- Create tables showing modal *Frequencies*, *Effective Mass Ratios*, and the sum of the *Effective Mass Ratios* for all translational and rotational directions
- Create a displacement magnitude overlay
- Animate the displacement overlay and review all of the calculated vibration modes

You will begin with a CAD file containing a solid model of a copper coil. The cross section of the coil is 2 mm square, it has an outside diameter of 32 mm, six full turns, and is 44 mm in length (including its terminations). You will calculate the first eight vibration modes.



2 - Set Up the Project

In this chapter, you will perform the following tasks:

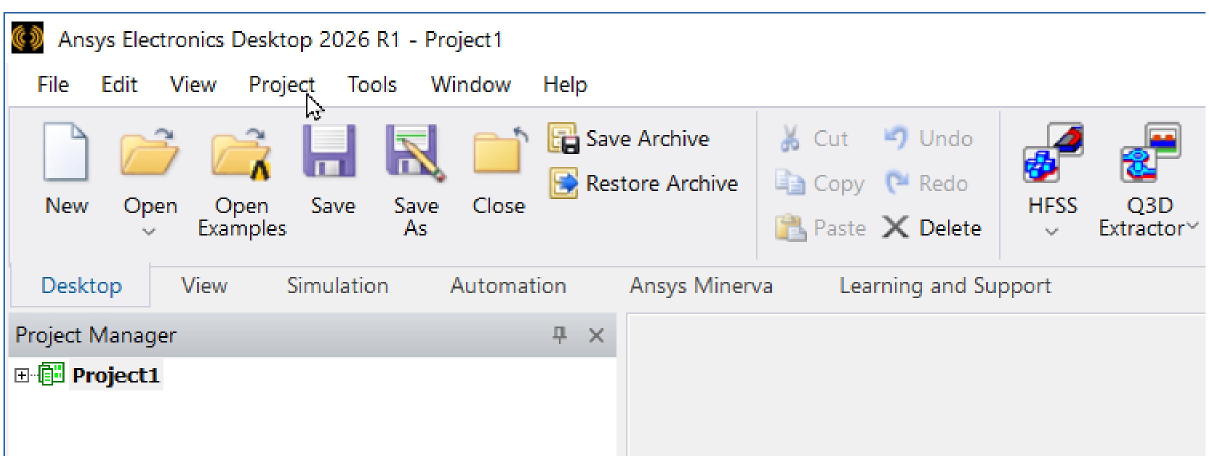
- Launch the Ansys Electronics Desktop application
- Insert an IcepakFEA design
- Choose the solution type (Modal)
- Set 3D UI options
- Choose the model units
- Import the CAD solid model
- Save the project to a working folder

Launch Ansys Electronics Desktop

For convenience, a shortcut to the Ansys Electronics Desktop (EDT) application is placed on your desktop during program installation. Optionally, you may want to pin the shortcut to your Windows Start Menu too. Before proceeding to the next topic, launch EDT and add a blank project, as follows:

1. Double-click the EDT **Ansys Electronics Desktop** shortcut on your desktop (or the same shortcut on your Start Menu).

   The Ansys Electronics Desktop application opens:



> **Note:**
>
> When you launch the application, a new, blank project is created automatically. However, if you were previously using the application and closed the project, you will have to create a new project by completing the following optional step.

2. If no project is listed in the Project Manager, click **New** on the **Desktop** ribbon tab.

   Project*x* appears at the top of the Project Manager.

Insert an IcepakFEA Design

Insert an IcepakFEA design and choose the Modal solution type, as follows:

1. On the **Desktop** ribbon tab, click **IcepakFEA**.

   The IcepakFEA design is listed beneath *Projectx* in the Project Manager:

   > **Note:**
   >
   > The default solution type for IcepakFEA designs is *Thermal*. However, you can change the default choice to the type you use most frequently. In the next step, you will determine the current *Solution Type* and change it if necessary.

2. If the design is not listed as **IcepakFEADesign*x* (Modal)** in the Project Manager use the menu bar and click **IcepakFEA** > **Solution Type**. Then:
   a. Select **Modal** from the *Solution Type* dialog box that appears.
   b. Click **OK**.

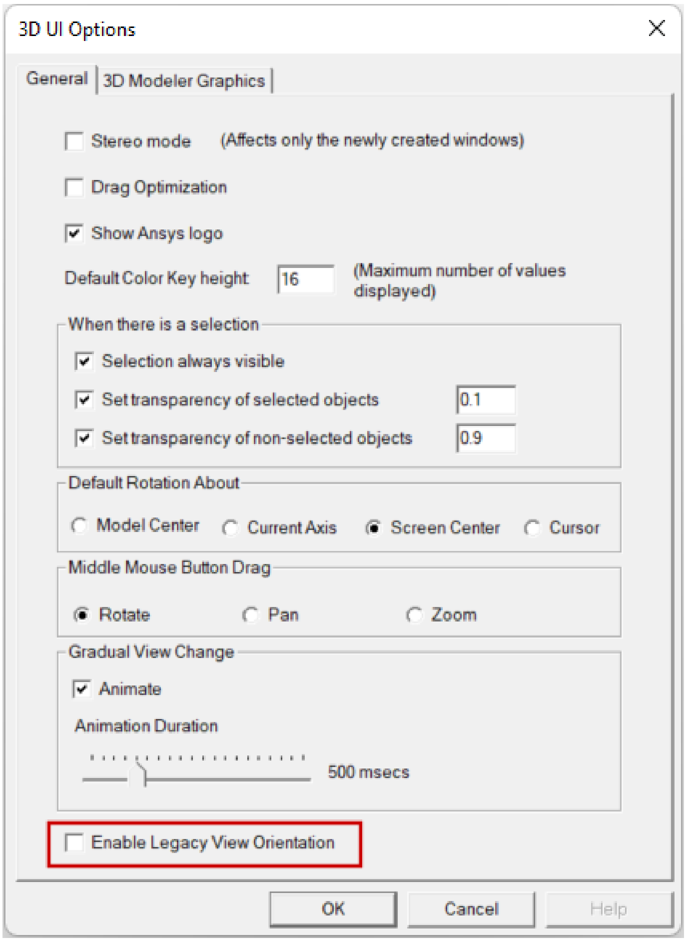
Set 3D UI Options and Model Units

Ensure that the legacy view orientation scheme is not being used, since the instructions and images in this guide are based on the new view orientation scheme introduced in release 2024 R1. Also, ensure that the selected model length unit is millimeters (mm):

1. From the menu bar, click **View** > **Options**.

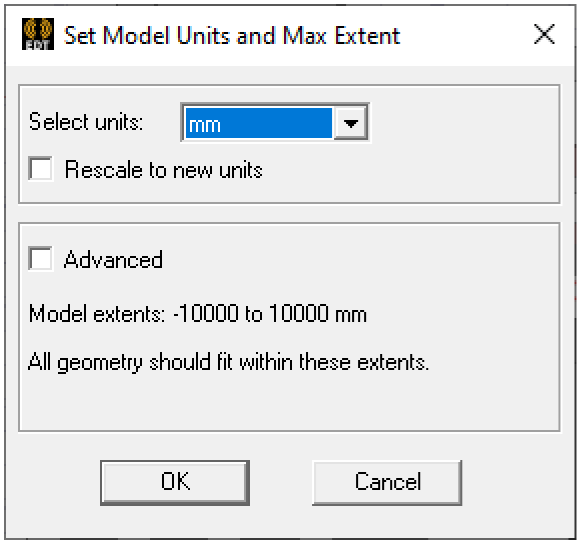
   The *3D UI Options* dialog box appears.

2. Ensure that **Enable Legacy View Orientation** is cleared:



3. Click **OK**.
4. On the **Draw** ribbon tab, click **Units**. (There is no icon associated with this command.) Then, in the *Set Model Units and Max Extent* dialog box, do the following:
    a. Ensure that **mm** (millimeters) is selected from the **Select Units** drop-down menu.

       Keep the *Rescale to new units* and *Advanced* options cleared.



b. Click **OK**.

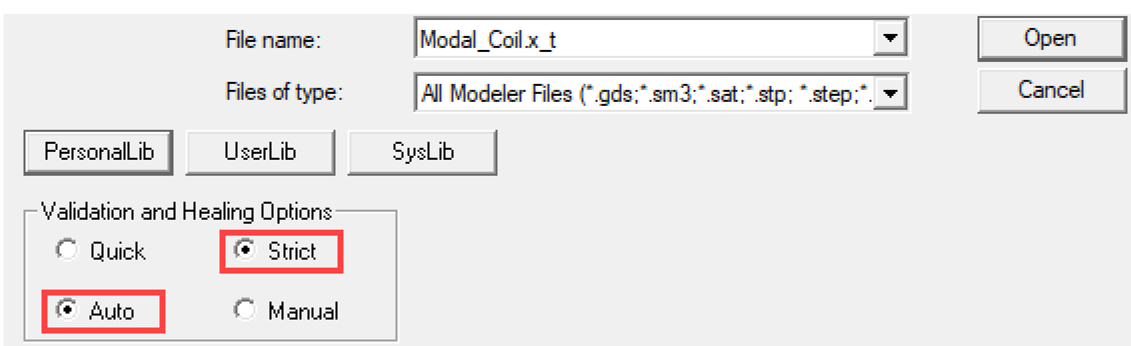
Import CAD Model

Import the CAD model to be used as the geometry source for this analysis, as follows:

1. Using the menu bar, click **Modeler** > **Import**.

   The *Import File* dialog box appears.

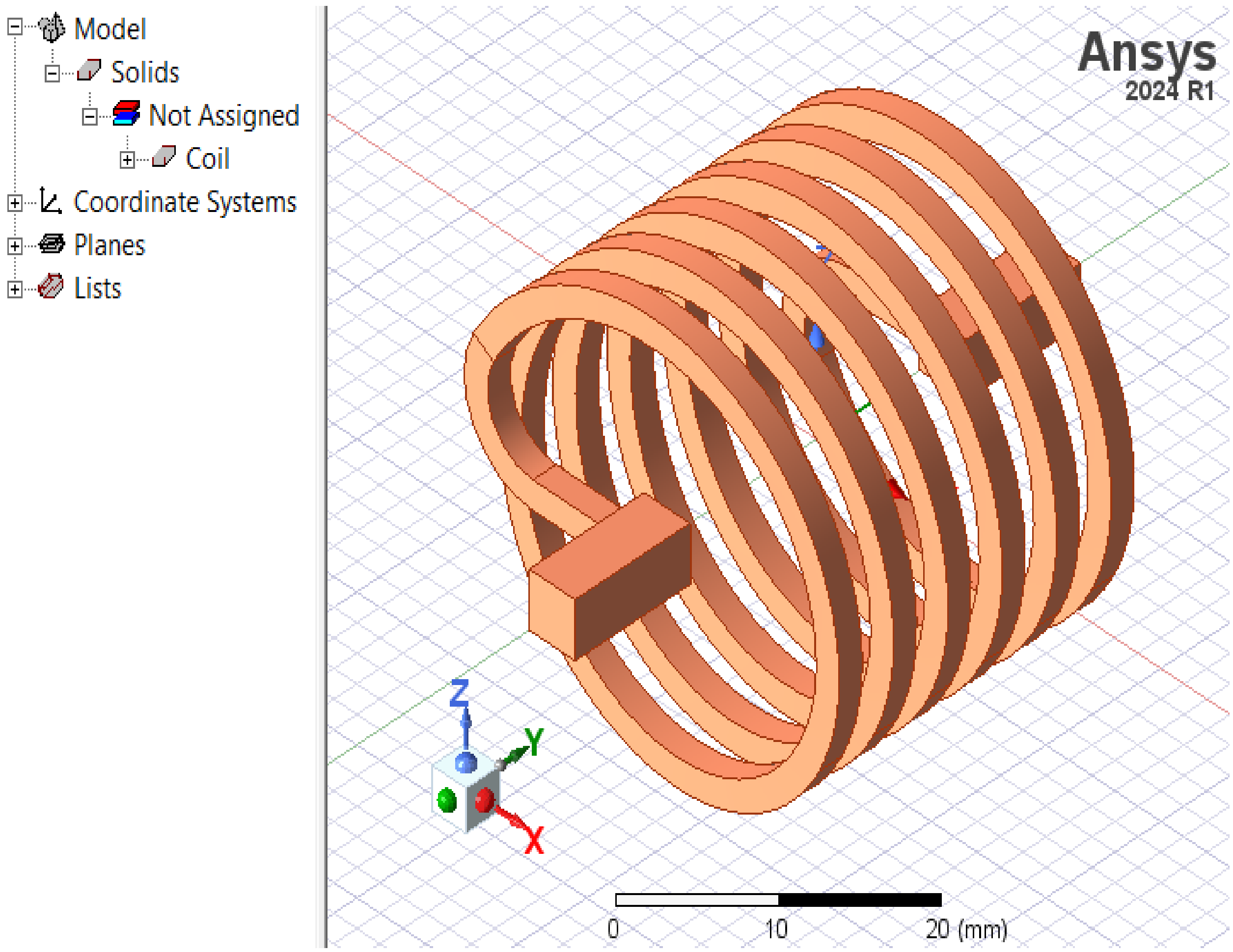
2. Navigate to the Ansys Electromagnetics Suite installation folder (typically "C:\Program Files\ANSYS Inc\v261\AnsysEM").
3. From this folder, drill down to the subfolder **\Help\IcepakFEA**.
4. Select the file **Modal_Coil.x_t**, which is in the Parasolid (text) CAD format. Then:
   a. Under *Validation and Healing Options*, select **Strict** and **Auto**.



b. Click **Open**.

The geometry appears in the Modeler window.

5. Click in the Modeler window's background area to clear the geometry selection.



Since you will not be creating any additional geometry during this exercise, hide the grid, ruler, and coordinate system axes. The *View Orientation Gadget* will remain visible and can be used as a reference for the direction of displacements and mass participation factors when evaluating the solution results.

6. On the **Draw** ribbon tab, click **Grid** to toggle off the grid visibility.
7. On the **Draw** ribbon tab, click **Ruler** to toggle off the ruler visibility.
8. Using the menu bar, click **View** > **Coordinate System** > **Hide**.

Finally, save the project to a suitable working folder and specify the file name, as follows:

9. On the **Desktop** ribbon tab, click **Save As**.
10. Navigate to a working folder of your choice. (Do not attempt to write to the program installation path.)

> **Note:**
>
> Optionally, you can click the **Create New Folder** icon ( ) within the *Save As* dialog box to create a new working folder in a suitable location.

11. In the **File name** text box, type **Modal_Coil** and click **Save**.

3 - Define Coil Material

The imported CAD solid body has no material assignment. Choose copper for the coil material as follows:

1. Under *Model* > *Solids* > *Not Assigned* in the History Tree, select **Coil**.

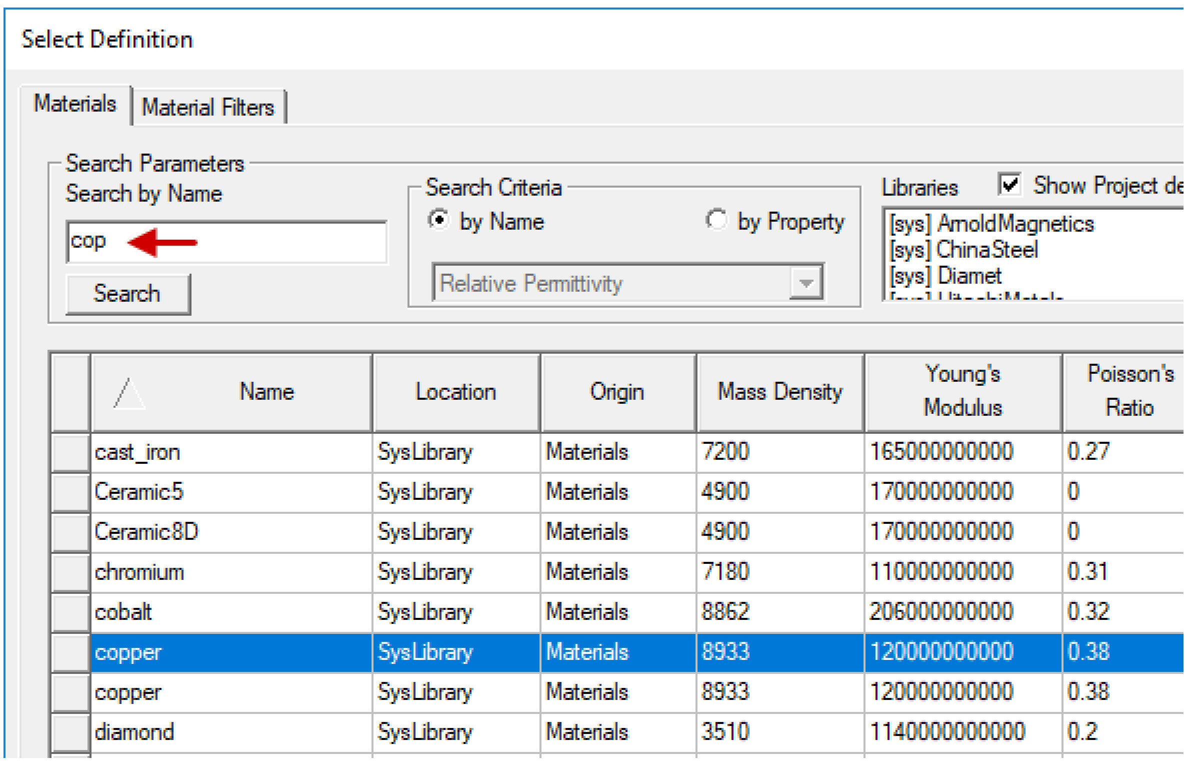
   The object's attributes appear in the docked *Properties* window.

2. Choose **Edit** from the **Material** drop-down menu in the *Value* column.

   The Select Definition dialog box appears.

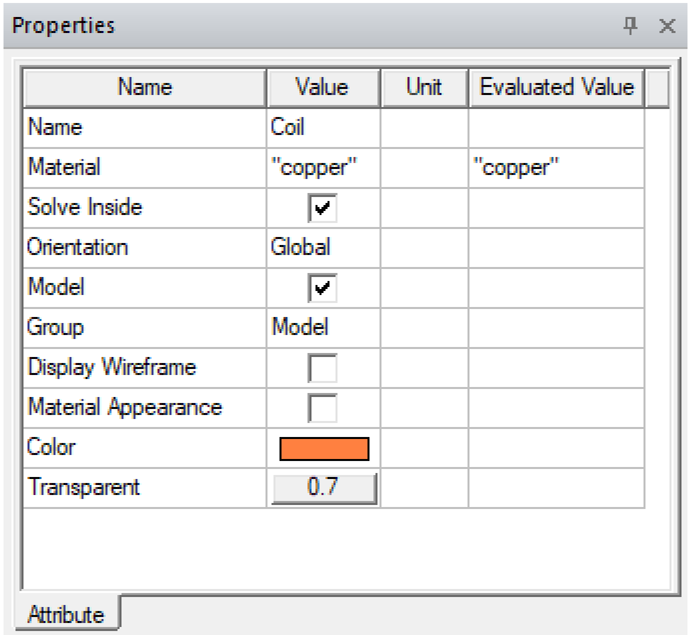
3. Type **cop** in the **Search by Name** text box.

   The material, **copper**, is located in the library and selected:



4. Click **OK** to apply this material to the coil.
5. Ensure that the **Material Appearance** option is *not* selected.
6. Set the **Transparent** value to **0.7** (70% transparency) and keep the orange color defined in the imported CAD solid model. The images in this guide are based on these material appearance settings.

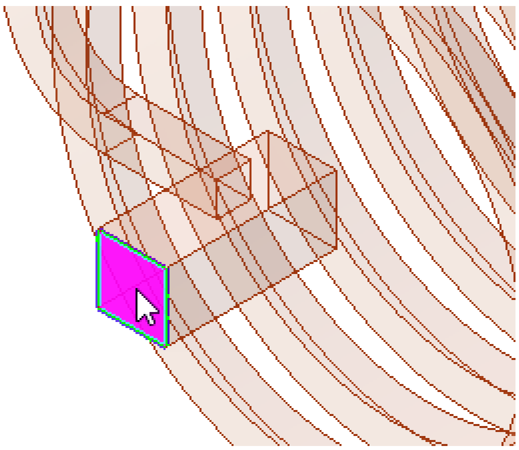
The *Attribute* tab of the docked *Properties* window should now match the following image:



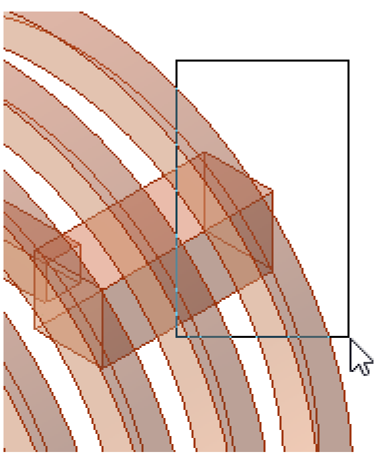
4 - Assign Boundaries

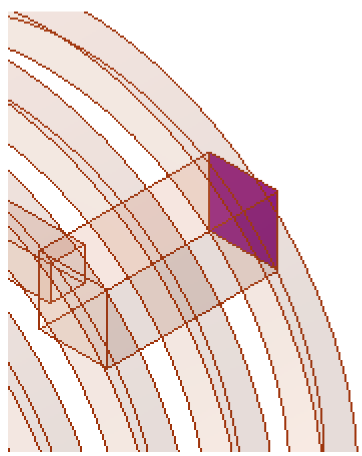
The coil is constrained at the square faces of the terminations at the far left and right ends of the model. You will apply a *Fixed Support* boundary at these two faces:

1. On the **Draw** ribbon tab, choose **Face** from the **Select** drop-down menu (or, with the Modeler window active, press **F** to begin the *Face* selection mode).
2. On the **Draw** ribbon tab, click **Orient** to ensure that you're looking at the default *Isometric* view of the model. (You do not have to access the *Orient* drop-down menu; the *Isometric* view is the default action.)
3. Click the face at the far *left* side of the model to select it:



4. Holding down **Ctrl**, click and drag (from left to right) to enclose only the small square face at the far *right* side of the model, selecting it also:

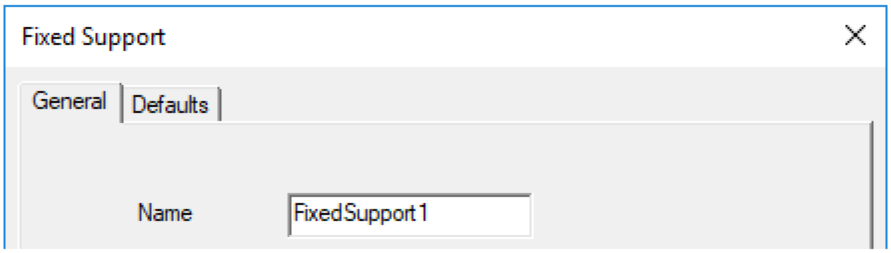




> **Note:**
>
> When dragging from left to right, only fully contained target entities (faces in this case) are selected. Dragging from right to left would define a *crossing* selection box, and every target entity that the box intersects or contains would be selected.

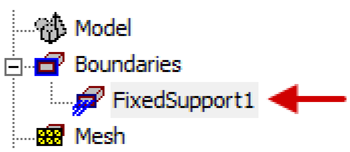
5. With both faces selected, right-click **Boundaries** in the Project Manager and choose **Assign** > **Fixed Support** from the shortcut menu.

   The *Fixed Support* dialog box appears:

6. Accept the default name and click **OK**.

   FixedSupport1 appears under *Boundaries* in the Project Manager:

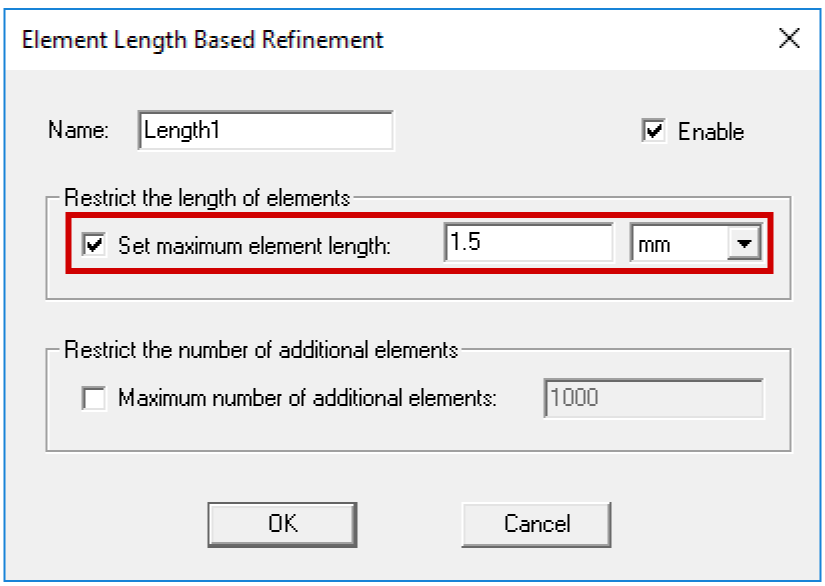
   

7. Click in the background area of the Modeler window to clear the selection.

5 - Assign Mesh Size

You will assign a length based mesh size inside the coil body to produce at least two elements through the thickness of the spiral portion.

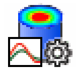
1. On the **Draw** ribbon tab, choose **Object** from the **Select** drop-down menu (or, with the Modeler window active, press **O** to return to the *Object* selection mode).
2. In the Modeler window, click anywhere on the coil to select it.
3. Right-click **Mesh** in the Project Manager and choose **Assign Mesh Operation** > **Inside Selection** > **Length Based** from the shortcut menu.
4. In the *Element Length Based Refinement* dialog box that appears, do the following:
    a. Ensure that the **Set maximum element length** option is selected.
    b. Type **1.5** in the text box to the right of this option.
    c. Choose **mm** from the adjacent drop-down menu, if this unit is not already selected.
    d. Click **OK**.



5.  **Save** your project. (This command is available from all ribbon tabs.)

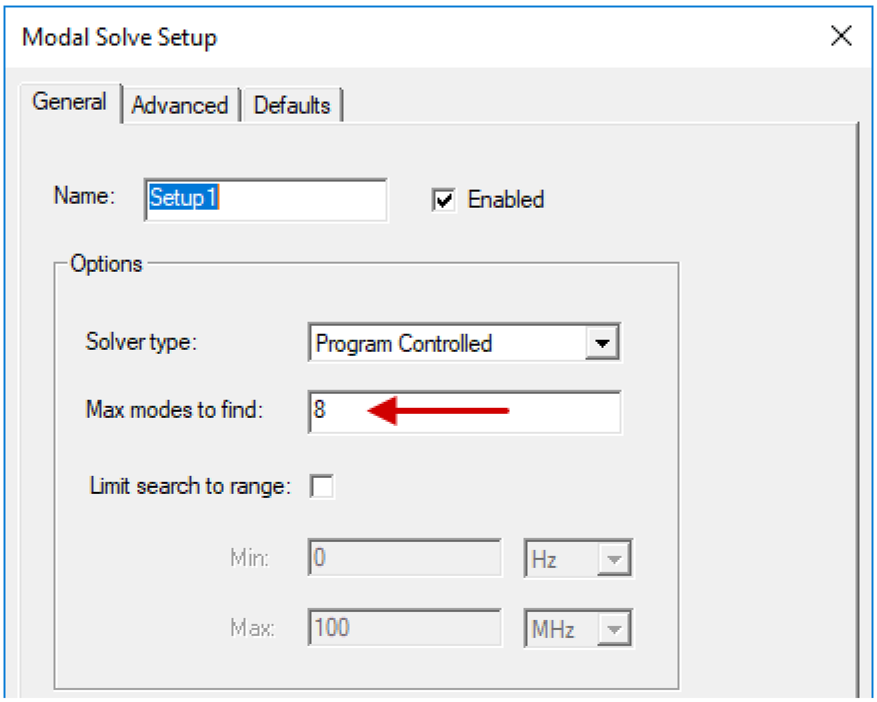
6 - Set Up, Validate, and Analyze Model

Specify the number of modes to calculate (8). Then, validate and run the analysis.

1. On the **Simulation** ribbon tab, click  **Setup**.

   The *Modal Solve Setup* dialog box appears.

2. In the **Max modes to find** text box, type **8**.

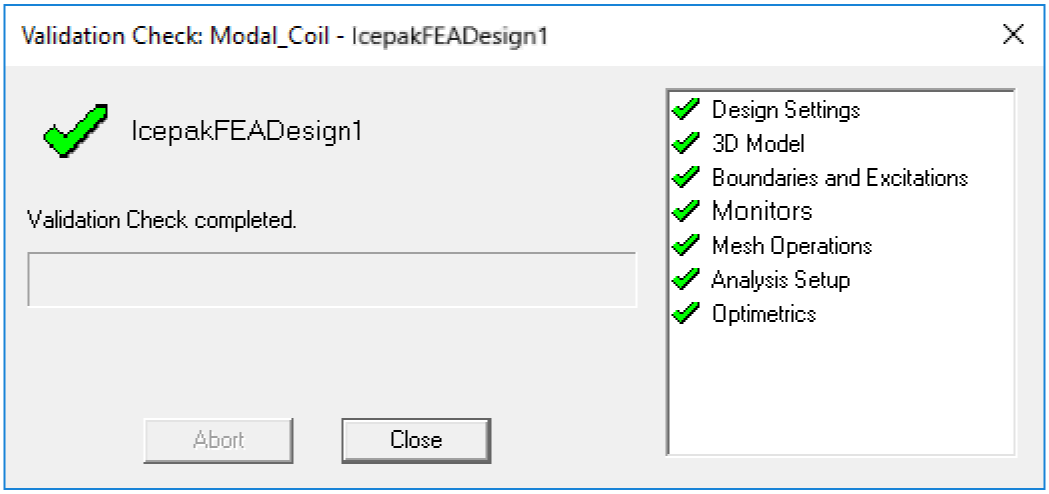


   Keep the default settings for the remaining options.

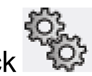
3. Click **OK**.

4. On the **Simulation** ribbon tab, click  **Validate**.

   The *Validation Check* window appears, and there should be no errors or warnings:



5. Click **Close** to dismiss the *Validation Check* window.

6. On the **Simulation** ribbon tab, click  **Analyze All**.

   The solution should take about two minutes to complete on a reasonably current computer workstation.

7 - Evaluate Results

In this final section, you will generate a data table and two field overlays to evaluate the mesh, tabulate the modal frequencies and effective mass ratios, and visualize the mode shapes.

The following three topics are covered in this section:

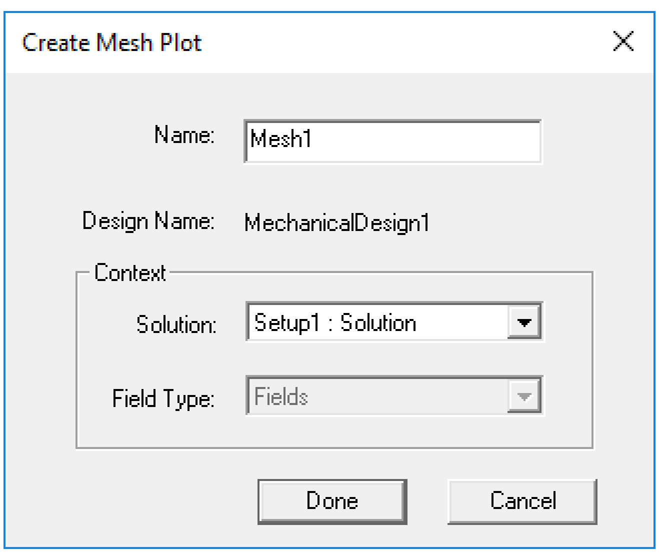
- **Create Mesh Overlay:** Display the generated elements on the undeformed faces of the model.
- **Create Modal Frequencies Table:** Produce a data table listing the eight natural vibration frequencies that were calculated along with the effective mass ratios in all six directions (x, y, and z translations and x, y, and z rotations).
- **Create and Animate Displacement Overlay**: Overlay the displacement magnitude results on a deformed representation of the model. Animate the overlay and review each of the eight mode shapes.

Create Mesh Overlay

A mesh overlay is best viewed on an undeformed view of the model. After creating the overlay, you will modify the *MeshPlots* attributes to turn off the *Scale Deformation* option.

1. In *Object* selection mode, click the **Coil** to select it. (You must select the geometry on which the requested result will be overlaid.)
2. Right-click **Field Overlays** in the Project manager and choose **Plot Mesh** from the shortcut menu.

   The *Create Mesh Plot* dialog box appears:

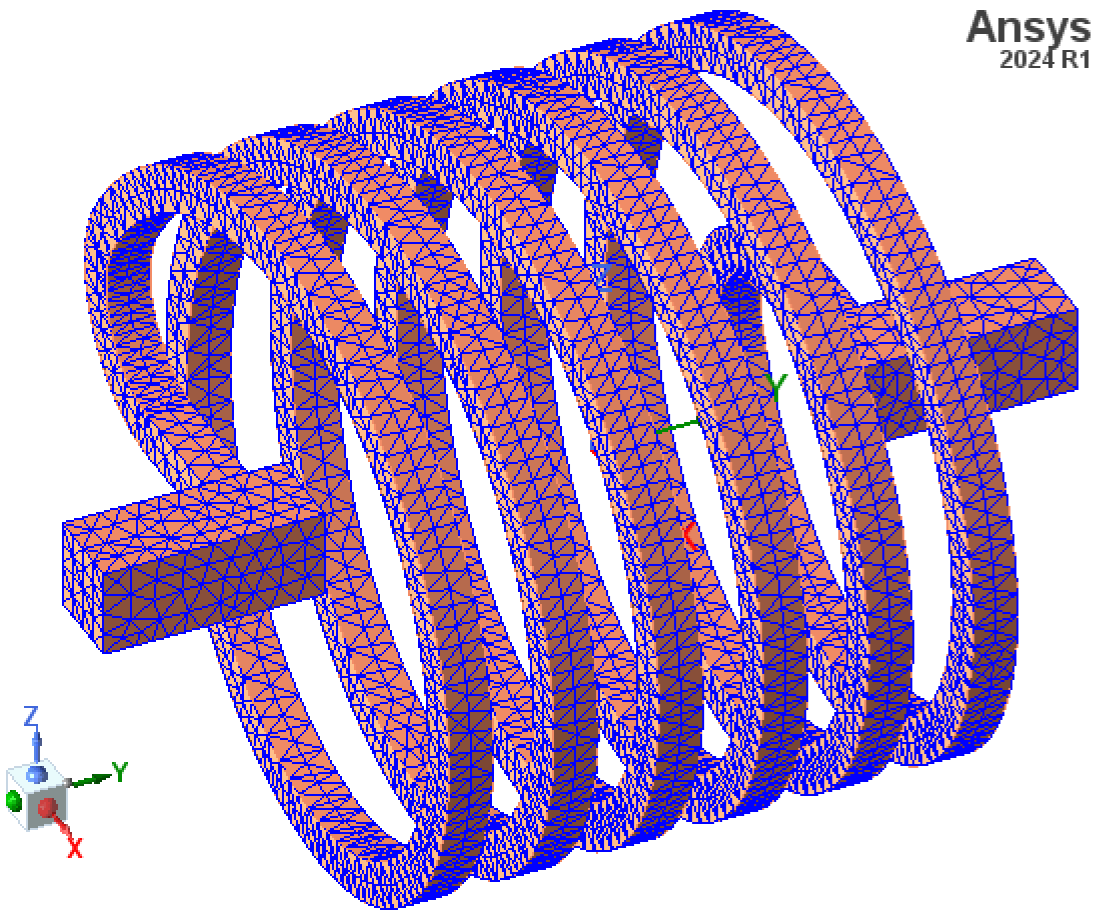
3. Click **Done** to accept the default plot setup and generate the mesh overlay.

4. Under **Field Overlays** in the Project Manager, right-click **MeshPlots** and choose **Modify Attributes**.
5. In the **MeshPlots** dialog box that appears, click **Deformation Scale**.
6. In the *Deformation Scale Factor* dialog box that appears, clear the **Scale Deformation** option and then click **OK**.
7. Click **Close** to dismiss the *MeshPlots* dialog box.
8. Press **F6** to turn off surface shading for the CAD geometry.

> **Note:**
>
> The mesh overlay shading will be based on the element faces rather than the original CAD model faces. This option improves the mesh display by preventing element edges from being obscured by the CAD surface rendering along curved faces.

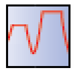
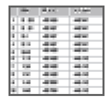
9. Holding down the **Z** key, click the **middle mouse button** and drag the cursor along an approximately circular path around the center of the model. A blue vertical line appears indicating the centerline of the constrained rotation axis (Z in this case). Red and Green crosshairs also appear along the line to indicate the rotation center point. Trace about a 25 to 30 degree clockwise arc about the rotation center point to produce a model viewpoint somewhere between the isometric and front views. This viewpoint will make the mesh lines along the OD and ID faces of the coil easier to see.

   Your model should now resemble the following image:


Ansys
2024 R1
Z
Y
X

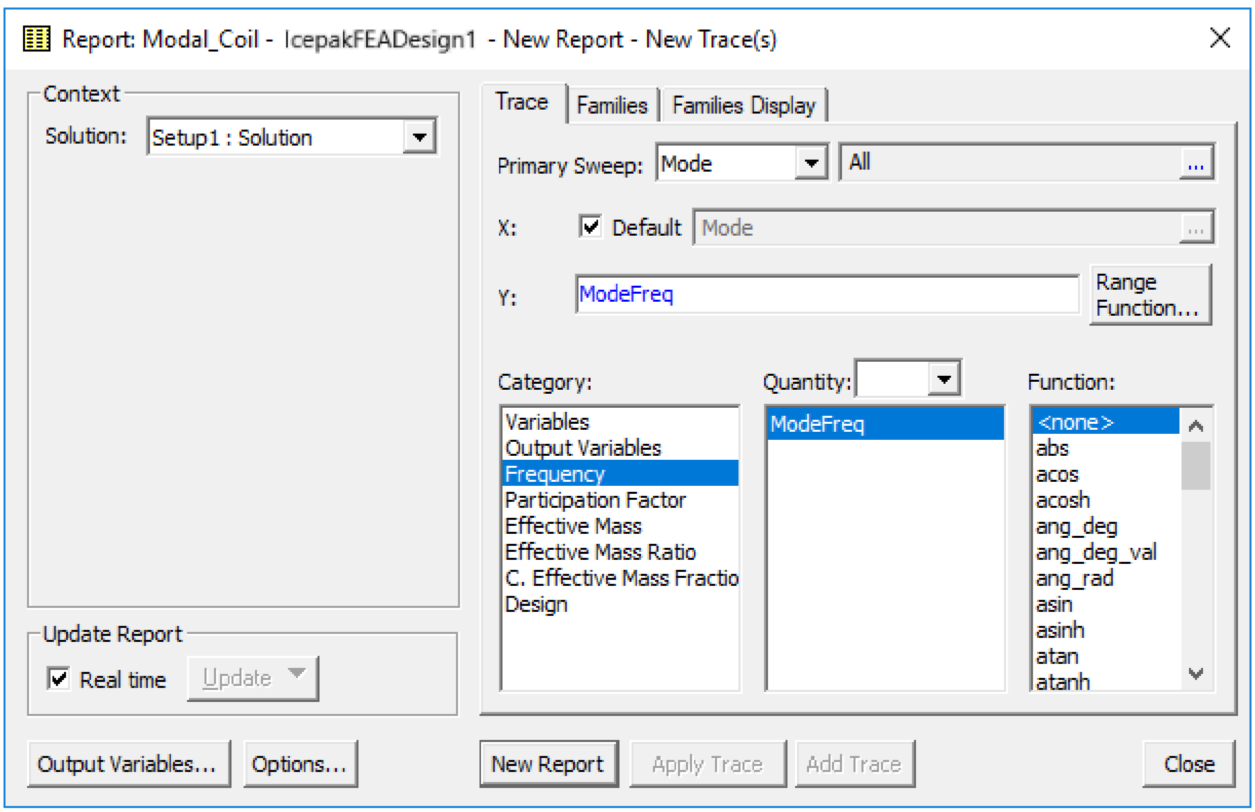
Create Data Tables

Modal frequencies and mass participation results are best reported in tabular form. In this procedure, you will create a data table showing the natural vibration frequencies for each calculated mode and the effective mass ratios for each mode in the three translational and three rotational directions. These ratios indicate the effective portion of the object's mass that participates in each vibration mode. A relatively high ratio indicates greater likelihood that an excitation at that frequency will cause structural resonance. Optionally, you will also create a table showing the sum of the effective mass ratios for the calculated modes.

1. On the **Results** ribbon tab, click  **Modal Report** >  **Data Table**.

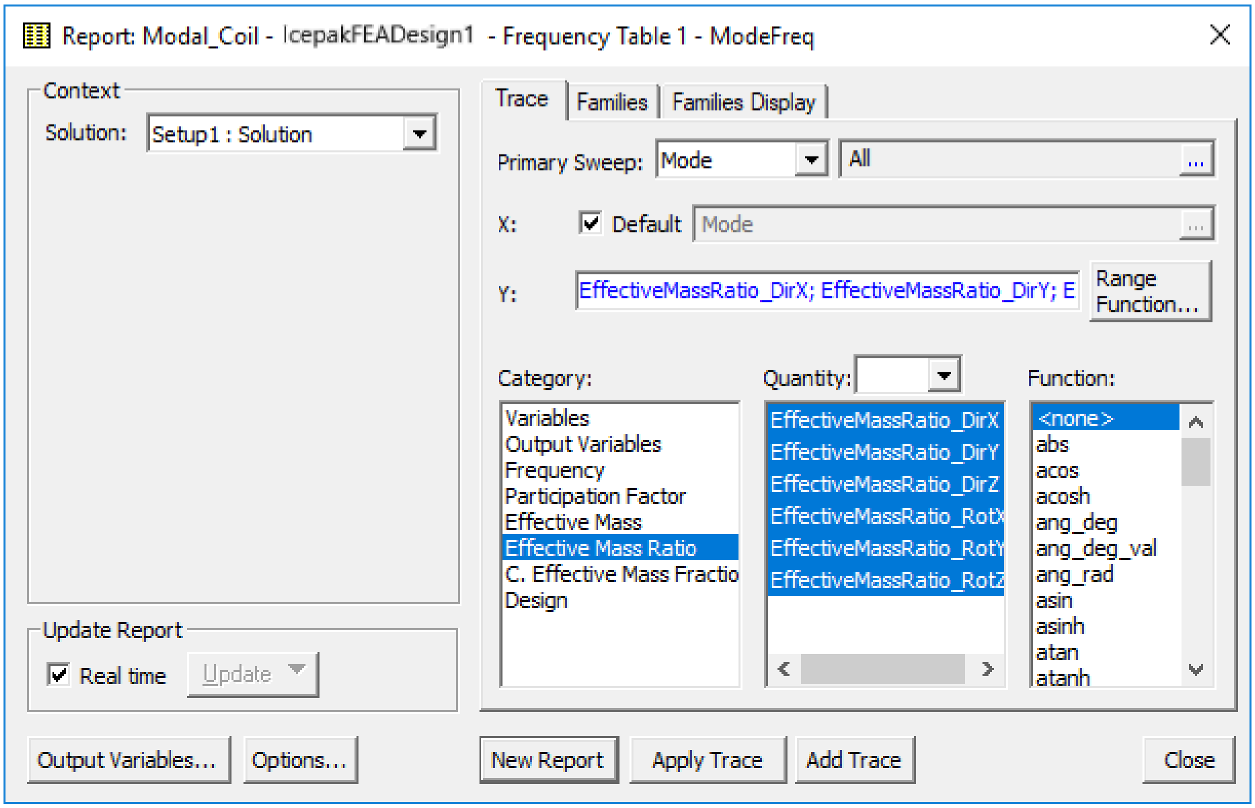
   Alternatively, right-click **Results** in the Project Manager and choose **Create Modal Report** > **Data Table** from the shortcut menu.

2. In the *Report* dialog box that appears, accept the default settings (as shown in the following image) and click **New Report**, but do not close the dialog box.



The data table appears in a new window.

3. In the *Report* dialog box that is still open, specify the following settings:
   a. Select **Effective Mass Ratio** in the **Category** list.
   b. Verify that **EffectiveMassRatio_DirX** is selected in the **Quantity** list.
   c. Holding **Shift**, click **EffectiveMassRatio_RotZ** in the **Quantity** list to also select all of the remaining quantities.

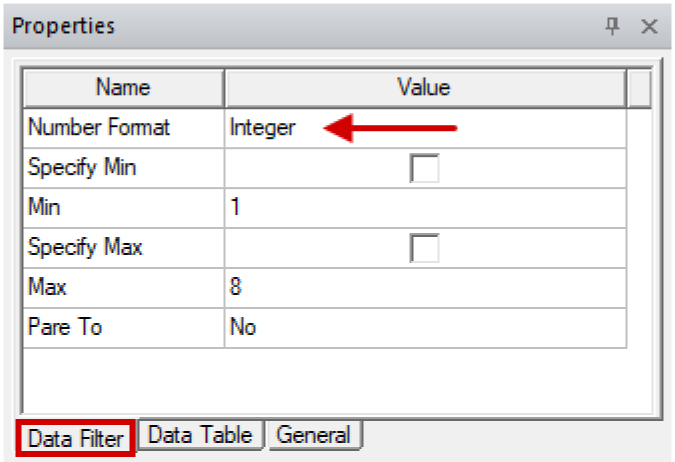


The *EffectiveMassRatio* columns are added to the table.

4. Click **Add Trace** and then click **Close**.
5. Click in the **Mode** column of *Frequency Table 1*.

   The column settings appear in the docked *Properties* window.

6. In the **Data Filter** tab of the docked *Properties* window, change the **Number Format** to **Integer**:



7. Click in the **ModeFreq** column of *Frequency Table 1*.
8. In the **Data Filter** tab of the docked *Properties* window, change the **Field Precision** value to **2**.
9. Click in the report window's background area to clear the current selection.
10. Click and drag the vertical column borders to size them as desired. You can also resize the report window.

    Your table should now resemble the following one:

Frequency Table 1

IcepakFEADesign1 **Ansys**

	Mode	ModeFreq [Hz] Setup1 : Solution	EffectiveMassRatio_DirX Setup1 : Solution	EffectiveMassRatio_DirY Setup1 : Solution	EffectiveMassRatio_DirZ Setup1 : Solution	EffectiveMassRatio_RotX Setup1 : Solution	EffectiveMassRatio_RotY Setup1 : Solution	EffectiveMassRatio_RotZ Setup1 : Solution
1	1	98.09	0.000000	0.671640	0.003016	0.000022	0.002402	0.000002
2	2	118.62	0.000000	0.002181	0.002611	0.000017	0.806081	0.007088
3	3	152.39	0.564342	0.000000	0.000055	0.016773	0.000000	0.003986
4	4	153.51	0.000054	0.001833	0.596866	0.004188	0.000205	0.001100
5	5	187.08	0.012885	0.000000	0.000001	0.191643	0.000000	0.000177
6	6	192.87	0.000000	0.000064	0.011171	0.000133	0.002064	0.451189
7	7	196.88	0.014380	0.000000	0.000000	0.272354	0.000000	0.000109
8	8	232.05	0.001393	0.000000	0.000000	0.004480	0.000000	0.000009

Optionally, complete the following additional procedure to produce a table of the effective mass ratio *sums*. Achieving a specified mass participation (for example, 85% or 90%) is a typical requirement when performing vibration analyses. If the minimum required mass participation is not achieved, the analyst can increase the number of modes to calculate in the solution setup and rerun the analysis.

11. On the **Results** ribbon tab, click **Modal Report** > **Data Table**.
12. In the *Report* dialog box, specify the following settings:
    a. Select **Effective Mass Ratio** in the **Category** list.
    b. Verify that **EffectiveMassRatio_DirX** is selected in the **Quantity** list.
    c. Holding **Shift**, click **EffectiveMassRatio_RotZ** in the **Quantity** list to also select all of the remaining quantities.
13. Click **Range Function**.
14. In the *Set Range Function* dialog box that appears, choose **sum** from the **Function** drop-down menu and then click **OK**.

    The contents of the **Y** text box have been modified to specify the *sum* of each ratio:

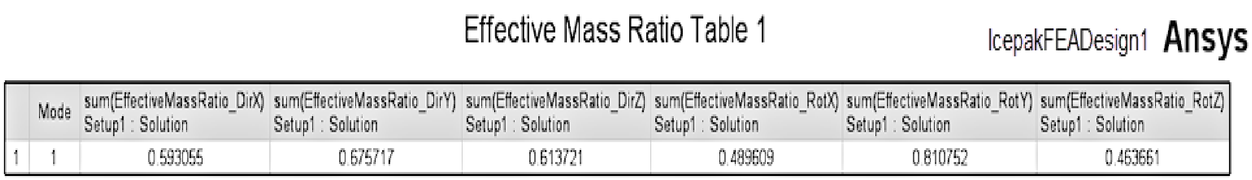
    

15. Click **New Report** and then click **Close**.

    Effective Mass Ratio Table 1 appears in a new window.

16. Under **Results** in the Project Manager, right-click **Effective Mass Ratio Table 1**, choose **Rename**, change the name to **Effective Mass Ratio Sums**, and press **Enter**.
17. As directed in step 6 above, change the **Mode** column's **Number Format** to **Integer**.
18. As directed in step 9 and 10 above, clear the selection and adjust the table column widths as desired.

    The revised table should resemble the following one:



Effective Mass Ratio Table 1

IcepakFEADesign1 Ansys

	Mode	sum(EffectiveMassRatio_DirX) Setup1 : Solution	sum(EffectiveMassRatio_DirY) Setup1 : Solution	sum(EffectiveMassRatio_DirZ) Setup1 : Solution	sum(EffectiveMassRatio_RotX) Setup1 : Solution	sum(EffectiveMassRatio_RotY) Setup1 : Solution	sum(EffectiveMassRatio_RotZ) Setup1 : Solution
1	1	0.593055	0.675717	0.613721	0.489609	0.810752	0.463661

Create and Animate Displacement Overlay

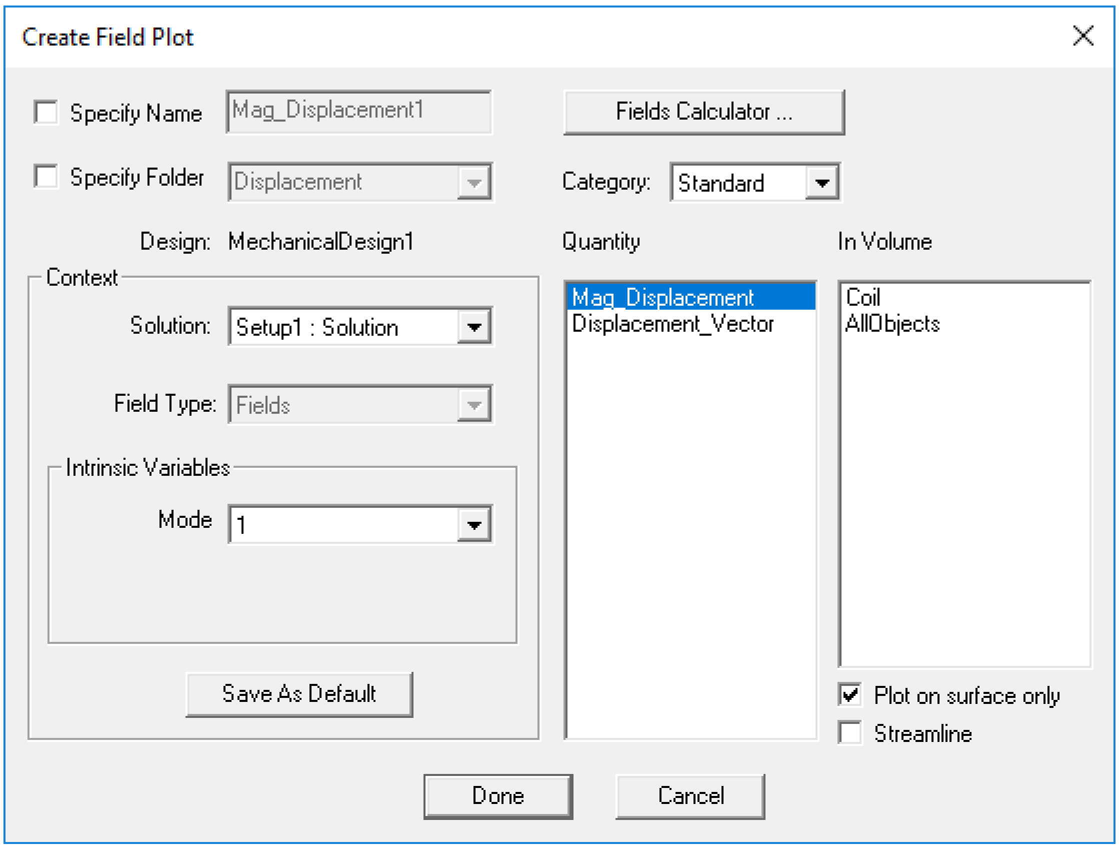
Finally, create an overlay of the displacement magnitudes, animate the overlay, and visualize the various mode shapes, as follows:

1. From the menu bar, click **Window** > **1 Modal_Coil - IcepakFEADesign1 - Modeler** to make the *Modeler* window the active window and to bring it to the foreground.
2. On the **Draw** ribbon tab, click **Orient** to restore the default *Isometric* view of the model.

3. Under **Field Overlays** > **MeshPlots** in the Project Manager, right-click **Mesh1** and clear the **Plot Visibility** option.

   Hiding the mesh will provide cleaner displacement visualizations.

4. In *Object* selection mode, click the **Coil** to select it.
5. In the Project Manager, right-click **Field Overlays** and choose **Plot Fields** > **Mag_Displacement** from the shortcut menu.
6. Verify that the settings in the *Create Field Plot* dialog box that appears match the following image:



7. Click **Done**.

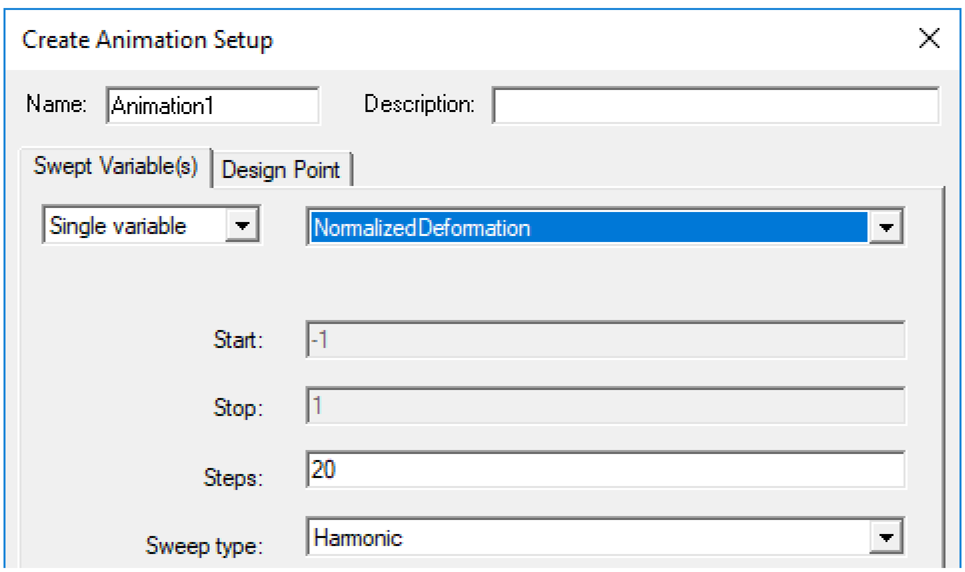
   The color contour overlay and displacement legend appear in the Modeler window. The displacement contour is displayed as the deformed shape of the coil.

Note:

The absolute magnitudes of the displacement results are arbitrary and have no meaning or significance because they are not scaled to any particular vibration excitation magnitude. Only the relative displacements are meaningful in that they provide an indication of the shape of the vibration mode. Do not be concerned that the displacement magnitude scale in the legend is not in proportion to the model size. The deformation scale is automatically adjusted to produce a reasonable deformed shape representation.

To better visualize the shape of the vibration mode, animate the overlay, as follows:

8. Under **Field Overlays** > **Displacement** in the Project Manager, right-click **Mag_Displacement1** and choose **Animate**.
9. In the *Create Animation Setup* dialog box that appears, type **20** in the **Steps** text box for a smoother animation:

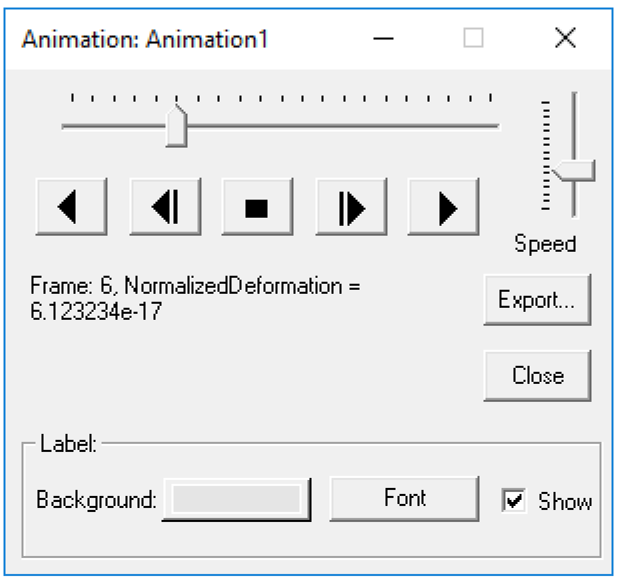


> **Note:**
>
> The *Harmonic Sweep type* is preferred for modal displacement overlays. The displacement varies sinusoidally from the calculated deformed shape to its inverse and then back to the calculated deformed shape. In this way, the full 360° vibration cycle is visualized. For example, an initially positive X displacement passes through zero and becomes a negative X displacement of the same magnitude at 180°. Then it cycles through zero again and back to the calculated deformed shape, completing the 360° vibration cycle.

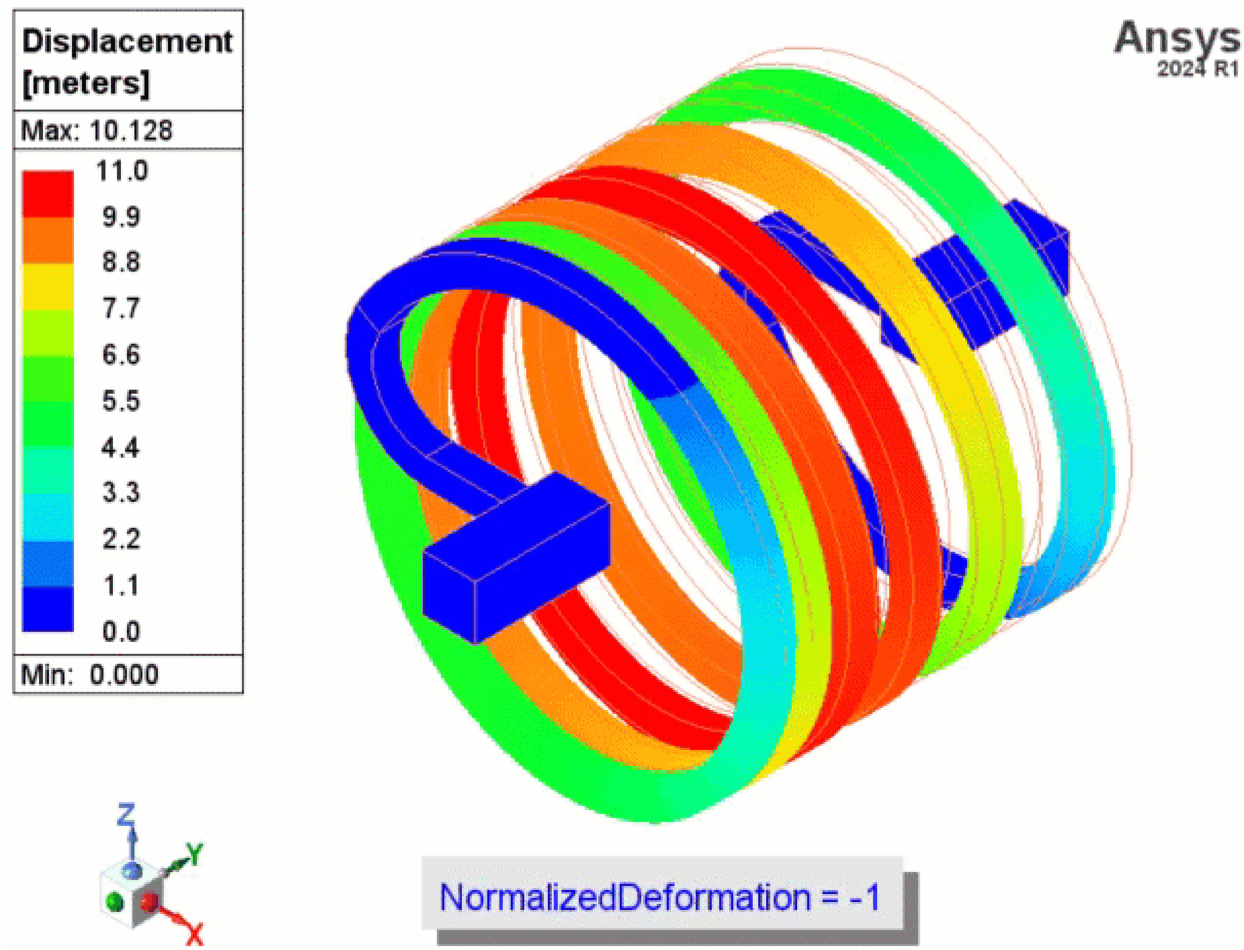
10. Click **OK**.

    The frames are calculated, the *Animation* dialog box appears, and the animation begins to play.

11. Use the controls in the Animation dialog box to stop, resume, select a particular frame, or change the speed of the animation:



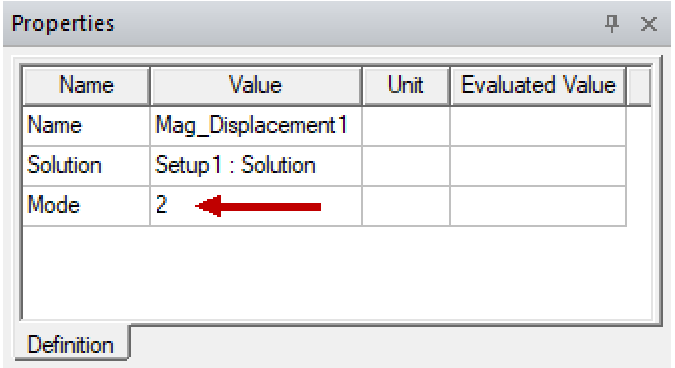
The first mode is an axial vibration mode. That is, it involves mostly Y translation, as shown in the following animation:



Notice that the normalized deformation magnitude is shown with the animation. That is, the displacement is scaled so that it cycles between 1 and -1 length unit.

Keep the animation playing as you continue through a review of the other natural vibration modes.

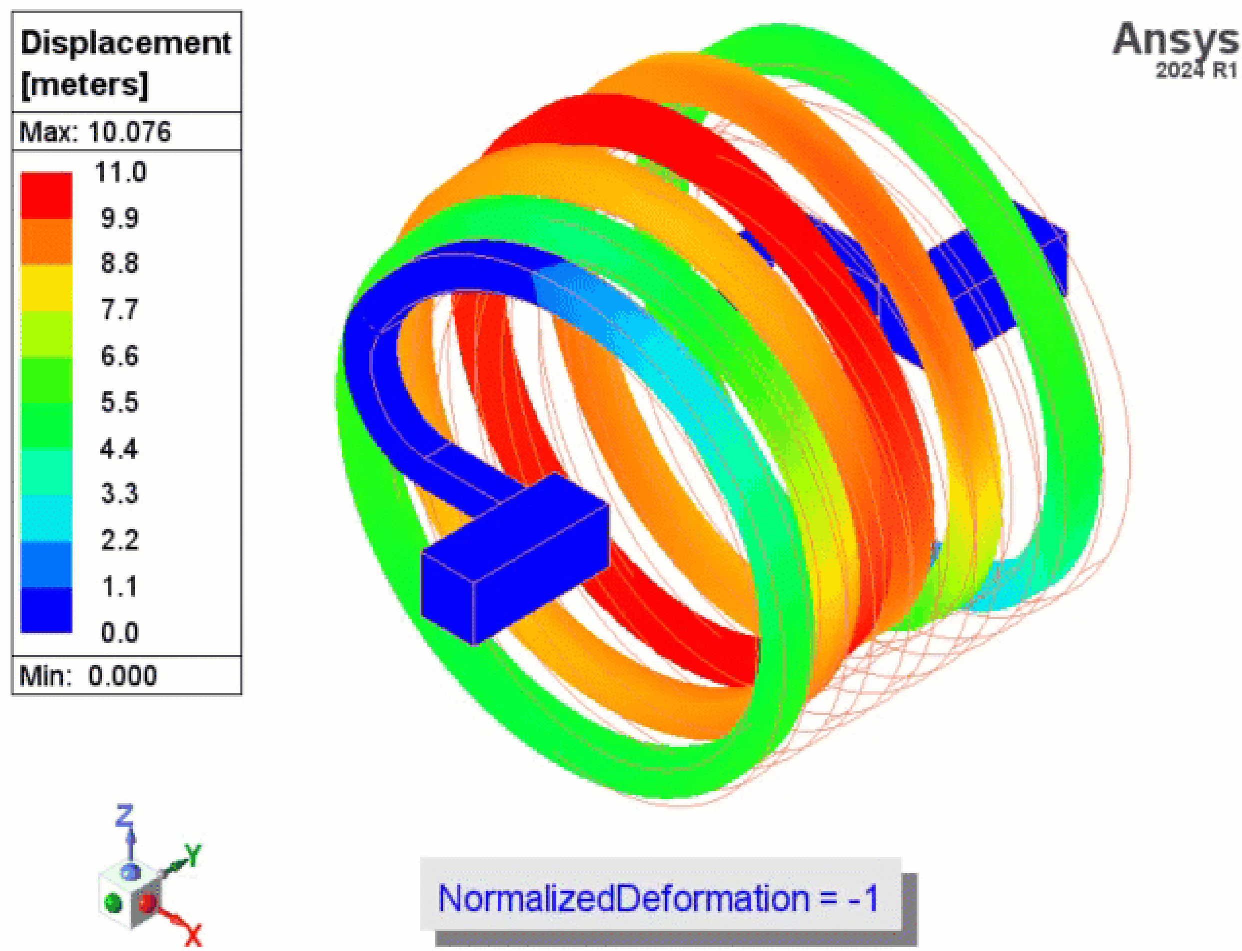
12. The displacement overlay definition should currently be displayed in the docked *Properties* window. If not, select **Mag_Displacement1** under **Field Overlays** > **Displacement** in the Project Manager.
13. In the **Definition** tab of the docked *Properties* window, change the **Mode** value from *1* to **2** and press **Enter** to observe the second mode shape animation:



The second mode is torsional, primarily involving rotation about the Y axis.

14. Change to **Mode 3**.

    This mode primarily involves X translation:



15. Change to **Mode 4**.

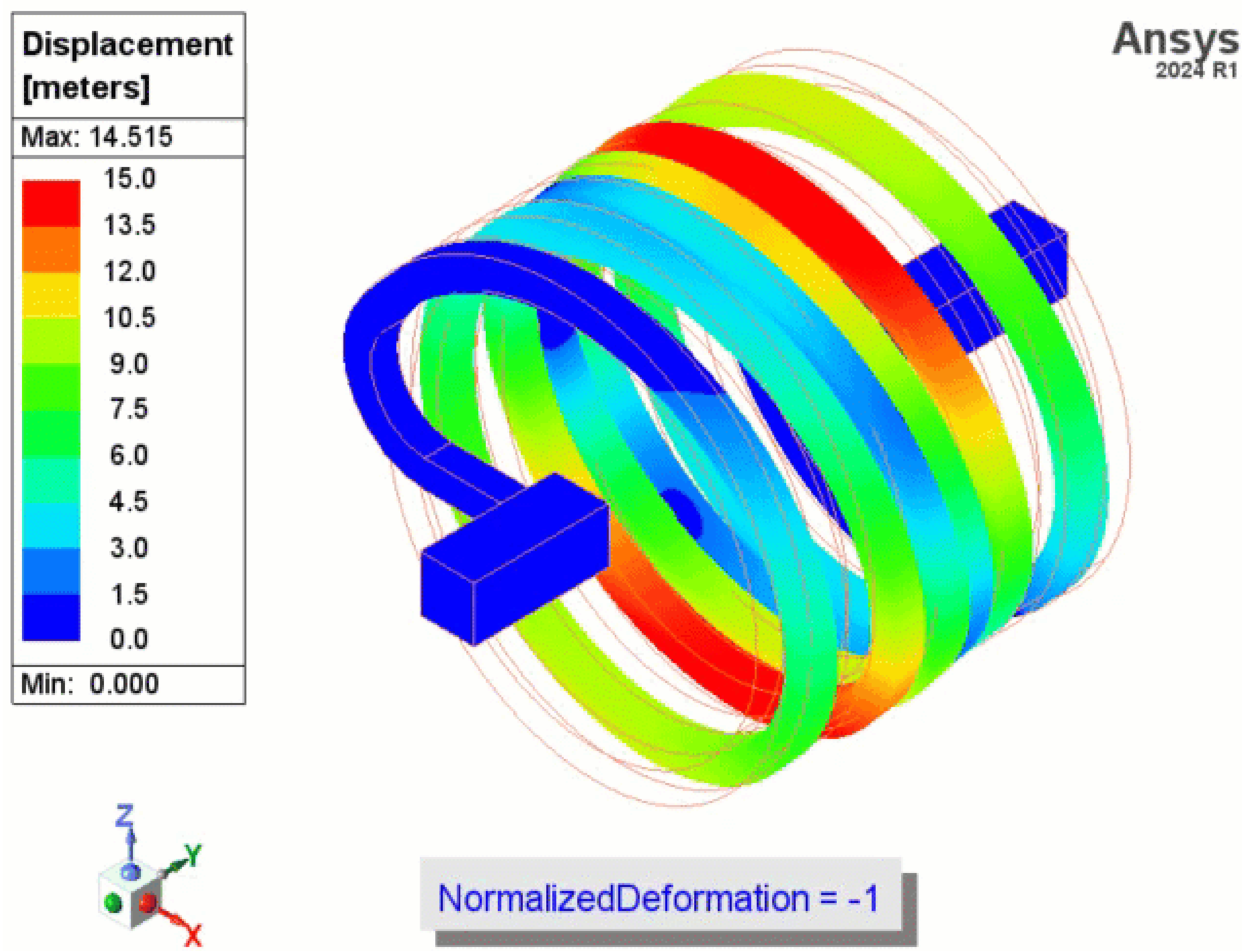
This mode is similar to *Mode 3* but this time the motion is in the Z direction. Not surprisingly, their frequencies are very close to each other, as tabulated in *Frequency Table 1*.

16. Change to **Mode 5**.

    This mode primarily involves X rotation.

17. If the turns of the coil appear to be overlapping each other, reduce the deformation scale as follows:
    a. Under **Field Overlays** in the Project Manager, right-click **Displacement** and choose **Modify Attributes**.
    b. Select the **Deformation Scale** tab of the *Displacement* dialog box that appears.
    c. Drag the **Scale Factor** slider to the left to reduce the active deformation scale.
    d. Click **Close** to dismiss the *Displacement* dialog box.

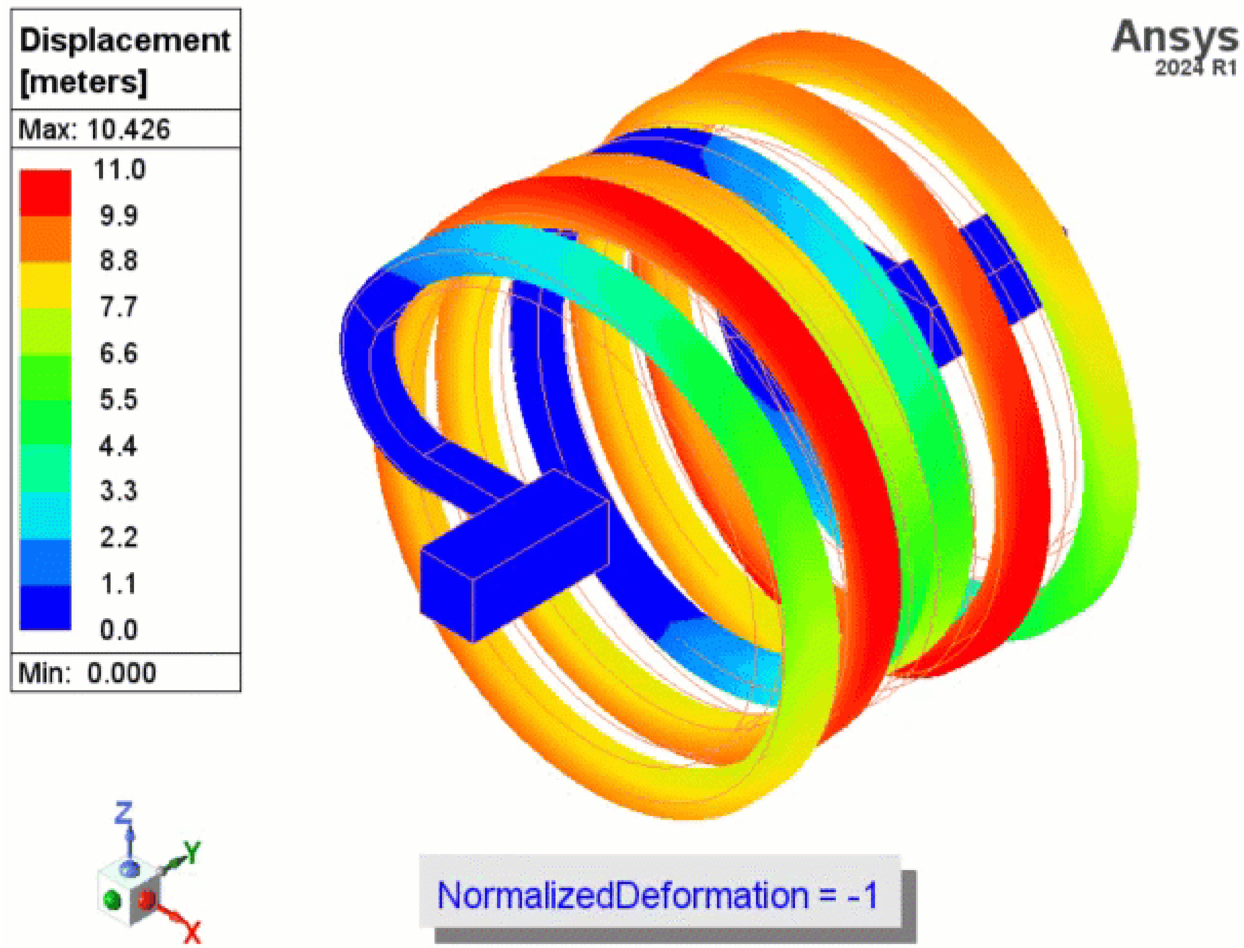
    The fifth vibration mode should resemble the following animation:



18. In the same manner, examine the sixth, seventh, and eighth vibration modes.

Observations:

- The sixth mode is similar to the fifth mode but this time, it primarily involves Z rotation.
- The seventh mode is similar to the fifth mode but involves fewer turns of the coil.
- The eighth mode is weak (as evidenced by the low tabulated effective mass ratios) and might be a harmonic of the second mode. There is torsional motion but with different turns of the coil rotating about the Y axis in opposing directions. The resultant effect is a complex mode shape involving a combination of Y rotation, Z rotation, X translation, and Y translation:



19. **Close** the Animation dialog box.

20. **Save** your project.

Congratulations, you have completed the *Modal Solution – Coil* getting started guide and can now close the project.